

Example 1c: Tensile Response of Monolithic Ti-21S

This example problem demonstrates MAC/GMC's capability to apply a loading history to a material and determine the response. For the sake of simplicity, the analyzed material is not a composite, but rather simply a monolithic titanium alloy commonly used in metal matrix composites; Ti-21S. Unlike the previous examples, the material properties for the Ti-21S are taken from the code's internal material property database. An applied strain of 0.02 (i.e., 2%) is applied to the alloy over 200 seconds (sec.), resulting in an applied strain rate of 10^{-4} /sec. This example illustrates the basic form of a MAC/GMC 4.0 input file involving applied mechanical loading and also could be of interest when the response of a particular constituent material is desired.

MAC/GMC Input File: `example_1c.mac`

MAC/GMC 4.0 Example 1c - monolithic Ti-21S

```
*CONSTITUENTS
  NMATS=1
  M=1 CMOD=4 TREF=23. MATID=A
#  M=1 CMOD=4 TREF=650. MATID=A
*RUC
  MOD=1 M=1
*MECH
  LOP=1
  NPT=2 TI=0.,200. MAG=0.,0.02 MODE=1
*SOLVER
  METHOD=1 NPT=2 TI=0.,200. STP=1.
*PRINT
  NPL=6
*XYPLOT
  FREQ=5
  MACRO=1
  NAME=example_1c X=1 Y=7
  MICRO=0
*END
```

Annotated Input Data

1) Flags: None

2) Constituent materials (***CONSTITUENTS**) [KM_2]:

```
NMATS=1
M=1 CMOD=4 TREF=23. MATID=A
#  M=1 CMOD=4 TREF=650. MATID=A
```

Number of materials:	1	(NMATS=1)
Constitutive model:	Isotropic GVIPS	(CMOD=4)
Material:	Ti-21S	(MATID=A)
Reference Temperature:	23. °C	(TREF=23.)

The reference temperature for a particular constituent material indicates the temperature at which the code should evaluate the material properties for that material. This overrides the temperature-dependence of a material's properties and causes the code to use the temperature indicated by TREF no matter what temperature the simulation is at. TREF is also useful, as in the current example, when no thermal loading is included in the simulation. In this case, without TREF, the code would not know at what temperature to take the Ti-21S material properties (and an error would result) since the properties stored for Ti-21S in the internal material database vary with temperature. Note that the temperature units (°C) are dictated by those used in the internal material database, which are always °C.

In addition, the example input file contains the line:

```
# M=1 CMOD=4 TREF=650. MATID=A
```

Here, the “#” sign in the first column indicates that this line is a “comment”, i.e., it is ignored by the code when the input file is read. Thus, by uncommenting this line and commenting out the line above, it is possible to execute the exact same case using Ti-21S material properties at 650 °C rather than 23 °C. Results for both of these temperatures are given below.

3) Analysis type (*RUC) → Repeating Unit Cell Analysis [KM_3]:

```
MOD=1 M=1
```

Analysis model:	Monolithic material	(MOD=1)
Material assignment:	Ti-21S	(M=1)

By including *RUC within the input file, repeating unit cell analysis, which represents a continuum has been selected. In this example, a monolithic material is analyzed, and the material to be analyzed must be selected from the materials indicated in *CONSTITUENTS.

4) Loading:

a) Mechanical (*MECH) [KM_4]:

```
LOP=1
NPT=2 TI=0.,200. MAG=0.,0.02 MODE=1
```

Loading option:	1	(LOP=1)
Number of points:	2	(NPT=2)
Time points:	0., 200. sec.	(TI=0., 200.)
Load magnitudes:	0., 0.02	(MAG=0., 0.02)
Loading mode:	strain control	(MODE=1)

The loading option indicates the direction of the applied mechanical loading (see the MAC/GMC 4.0 Keyword Manual Section 4 for details on the loading options). In this case the mechanical loading is applied in the x_1 -direction, with all other stress components kept at zero. The mechanical loading profile is specified through time-magnitude pairs that specify the loading history to be applied (the first time must always be zero). In MAC/GMC 4.0, the unit of time is seconds (sec.). Finally, the loading mode indicates whether strains (MODE=1) or stresses (MODE=2) are applied. The applied mechanical load history for this problem is plotted in [Figure 1.3](#).

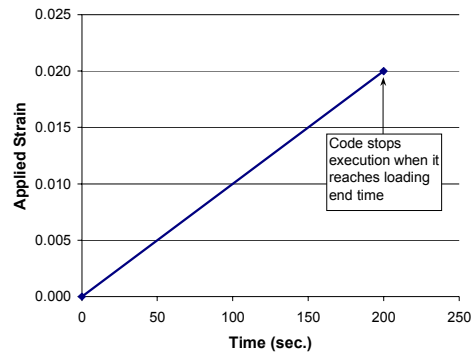


Figure 1.3 Example 1c: Applied strain vs. time history.

☞ Note: The applied global strain rate is $0.02 \div 200. \text{ sec.} = 1 \times 10^{-4} / \text{sec.}$

b) Thermal (***THERM**): None

No thermal loading is applied in this example. This necessitates the use of TREF in ***CONSTITUENTS** to indicate the temperature at which the material properties are taken. If a thermal loading profile were specified, TREF would not be needed (although it could be specified).

c) Time integration (***SOLVER**) [KM_4]:

```
METHOD=1 NPT=2 TI=0.,200. STP=1.
```

Time integration method:	Forward Euler	(METHOD=1)
Number of time points:	2	(NPT=2)
Time points:	0., 200. sec.	(TI=0., 200.)
Time step size (from 0. – 200. sec.):	1. sec.	(STP=1.)

Two explicit methods of time integration are available in MAC/GMC 4.0; Forward Euler (METHOD=1) with a specified time step and Predictor/Corrector (METHOD=2) with a self-adaptive time step size. In this example, the simpler Forward Euler method is employed. The forward Euler time integration is specified by a time point – time step size profile. Since the time step size pertains to a time segment between two time points, there will always be one fewer time step size specified than the number of time points specified. As in ***MECH**, the first time must always be zero.

5) Damage and Failure: None

6) Output:

a) Output file print level (***PRINT**) [KM_6]:

```
NPL=6
```

Print level:	6	(NPL=6)
--------------	---	---------

The print level indicates how much information will be written to the output file. A level of 0 prints the least amount of information while a level of 10 prints the most. A print level of 6 results in an intermediate level of writing to the output file including the effective stiffness matrix at each time step. See the MAC/GMC 4.0 Keyword Manual Section 6 for details on the various print levels.

b) x-y plots (***XYPLOT**) [KM_6]:

```
FREQ=5
MACRO=1
NAME=example_1c X=1 Y=7
MICRO=0
```

Frequency:	5	(FREQ=5)
Number of macro plots:	1	(MACRO=1)
Macro plot name:	example_1c	(NAME=example_1c)
Macro plot x-y quantities:	$\epsilon_{11}, \sigma_{11}$	(X=1 Y=7)
Number of micro plots:	0	(MICRO=0)

In this example, a macro (repeating unit cell level) plot file containing the σ_{11} - ϵ_{11} stress-strain response is generated as indicated by the macro plot x-y quantities X=1, Y=7 (see the MAC/GMC 4.0 Keyword Manual Section 6 for details on the macro plot x-y quantities). No micro (subcell level) plots are generated. The frequency indicates how often data will be written to the x-y plot data file. In this case, data will be written every 5 time steps (and thus every 5 seconds of the simulated applied loading). The extension “_macro.data” is appended to the indicated macro plot file name, so, in this example, the file “EXAMPLE_1c_macro.data” is written to the path location associated with the input file.

7) End of file keyword: (***END**)

Results

Results are written to both the output file and the x-y plot file in this example. The output file contains all input information regarding the case executed, as well as calculated effective properties and detailed information at each time step in the loading profile (depending on the print level indicated in *PRINT). It is a good idea to check the output file for warnings, which can indicate a problem or unintended result of the input data specified. In the present example, the code has generated two warnings based on the use of TREF and the lack of thermal loading information. The data written to the x-y plot file have been plotted in [Figure 1.4](#). Clearly, the Ti-21S stress-strain response exhibits a significant effect of temperature. Given temperature dependent material properties, MAC/GMC 4.0 captures the temperature dependence in the results, whether for a monolithic material (as in the present case) or for a composite (see Example 1d).

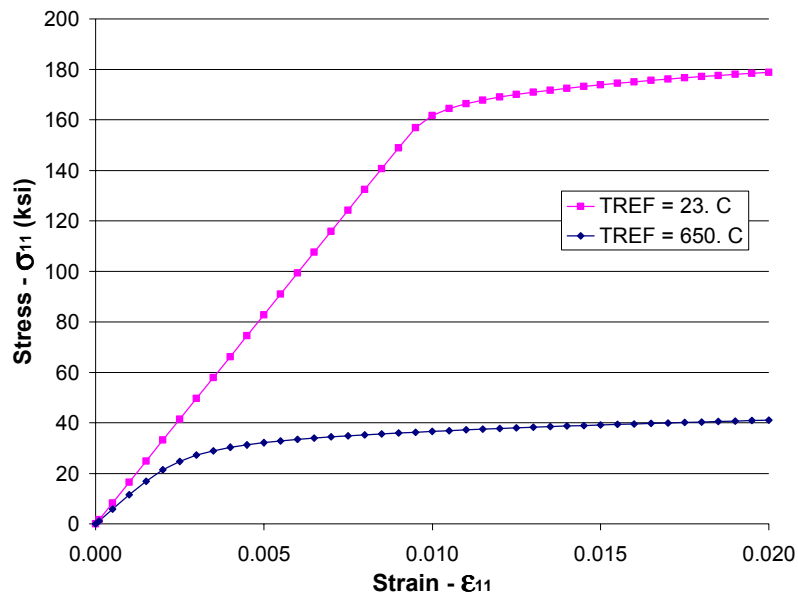


Figure 1.4 Example 1c: plot of the simulated σ_{11} - ϵ_{11} stress-strain response of Ti-21S at 23. °C and 650 °C. Note that the 650 °C results were generated by commenting and uncommenting the appropriate lines in the input file under ***CONSTITUENTS**. Note that the global applied strain rate in this example is 1×10^{-4} /sec.